공학석사학위논문

Enhancing Bubble Behavior on Heated Surface with External Flow using Volume of Fluid Model

유체적 모형을 이용한 외부유동 가열면 기포거동 증진

2014 년 2 월

서 울 대 학 교 대 학 원
에너지시스템공학부
정 찬 희

Enhancing Bubble Behavior on Heated Surface with External Flow using Volume of Fluid Model

유체적 모형을 이용한 외부유동 가열면 기포거동 증진

지도교수 서 균 렬

이 논문을 공학석사학위 논문으로 제출함

2014 년 2 월

서 울 대 학 교 대 학 원

에너지시스템공학부

정 찬 희

정찬희의 석사학위 논문을 인준함

2014 년 2 월

위 원 장 조 형 규 (인)

부위원장 서 균 렬 (인)

위 원 김 응 수 (인)

ABSTRACT

In this study, we researched augmenting bubble behavior on heat with external flow using volume of fluid model. The subject is also concerned about the implementation methodology of heat transfer enhancement in nuclear power plants.

The main objective of this research is to show how force convection heat transfer can be augmented in nuclear power plants by means of a certain structure that can push hot liquids or bubbles out of neighboring heated surface. We carried out numerical experiments in this regard considering theoretical backgrounds and result have been elucidated in the subsequent sections. When an intricate piece of machinery of reactor core (or U-shaped heat exchanger) is taken into consideration, it is observed that an artificial flow of fluid is generated by an impeller or outlets of induction pipes that cannot be embodied easily in case of fission reactors, although it can be easily figured out in case of tokamaks.

The main goal of this study is how to use force convection for nuclear plants with a certain device and design plan. For example, in nuclear power plant, a structure which can push hot liquid or bubbles out of neighboring area of heated surface for force convection was bodied out. When an intricate piece of machinery of reactor core (/or U-shaped heat exchanger) is taken into consideration, it seems that an artificial flow of fluid which is produced from an impeller or outlets of induction pipes cannot be bodied out easily. But an artificial flow of fluid which is produced from an impeller or outlets of induction pipes can be bodied out even in tokamaks. In the case of structures which produce forced convection or bubble pushing, the volume of provided cooled liquid or the speed of departure of bubble on the surface of heated surface is expected to be in proportion to speed of moving structure, moving liquid or the effect of shape of volume.

Apart from that numerical study of the movement of bubble is performed using the Volume of Fluid (VOF) model of commercial CFD software named ANSYS. The modeling of force convection heat transfer in order to increase the departure of bubble is performed. Also, the behavior of the bubble which is elicited by the different type of force convention is investigated with expectation of effectiveness for heat transfer enhancement in nuclear plants.

As a method of enhanced heat transfer, this paper focused on the characteristics of convective heat transfer by using forced artificial flow which connected with motor or pump or outer force such as flow of fluid.

A bubble cutter system has been used as the passive heat exchanger in this study. In addition, an artificial flow system that can cut (and/or push and break) bubbles is explained to drag bubbles. The rate of convective heat transfer will be increased by the local forced convection.

In this study, different types of artificial flow have been examined by comparing results with VOF modeling. The departure of bubble can be accelerated by the artificial flow which in turn will increase heat transfer. The qualitative features of the bubble departure have been shown by numerical simulation.

Keywords: Heat Transfer, Bubble Behavior, Departure of Bubble, Nuclear Plants, Local Forced Convection, Volume of Fluid

Student ID: 2012-21010

CONTENTS

LIST OF TABLES

LIST OF FIGURES

1. INTRODUCTION

The heat transfer is an important parameter in the design and operation of nuclear reactor fuel rod assemblies in nuclear plants.

The enhanced heat transfer can convert heat more effectively from heat source. Especially, the critical heat flux is the condition in which the tube wall is not wetted by the liquid and is in direct contact with the vapor phase. The resulting deterioration of the convective boiling heat transfer mechanism leads to a large rise in the wall temperature in a heat flux imposed system such as a nuclear reactor. So, the enhanced critical heat flux which prevents a large rise in the wall temperature is demanded in the design.

Of course, the enhanced heat transfer is required in fusion reactor. For instance, it may permit wide range of performance of each component in the design of blankets of a fusion reactor. Furthermore, it can also allow fusion-fission hybrid system. For example, nuclear fusion by magnetic confinement has made great progress in recent times despite its commercial application is yet far away to achieve. So, the fusion hybrid can be considered as an alternative.

In the recent years, many researchers have attempted to increase heat transfer as well as Critical Heat flux (CHF) by using nano-particles and even by varying shape and arrangement of spacer grids. Researchers have been trying to analyze the bubble behavior experimentally and academically for the aims.

While the simplicity of the systems also implies economic construction and operation compared to other kinds of power plants, nuclear plants needs many bundle of pipes for heat transfer. Not only the enhanced heat transfer but also the enhanced safety is prerequisite for expanding market of the nuclear plant. The safety should be ensured concurrently, as we have got a living lesson in case of a loss-of-coolant accident, hence melt down accident must be prevented at any cost. When researcher analyzes that kind of important environment setting, the bubble behavior in various conditions must be emphasized with special attention. Therefore in order to increase heat transfer it may be necessary to manage the bubble behavior.

In order to meet these galloping demands of nuclear power industry, development of an innovative heat transfer methodology is of utmost importance which will be both economic and will achieve greater sustainability, safety and reliability.

2. RESEARCH TREND

2.1. Research Trend on Nuclear Plants

Enhanced heat transfer is one of the prospective aims for nuclear power plant (NPP). Regarding to the flow boiling heat transfer enhancement with surfactant solutions under atmospheric pressure, surfactant solutions or surface conditions have an effect on the behavior of occurrence bubbles on a heated surface.

Based on a number of theories related to the bubble dynamics and heat transfer, many works have explained how bubbles get loosed from its attachment to a heated surface [1]. The phenomenon is explained in Figure 2-1. It shows growth and departure of two bubbles for a wall superheat of 10°C taken from the reference [2]. The experimental and numerical methods are basically the same with little difference.

It is also investigated in a number of researches how enhanced heat transfer can be obtained using the bubbles behavior on nano coating layer can enhance the Critical Heat Flux (CHF) [3].

.

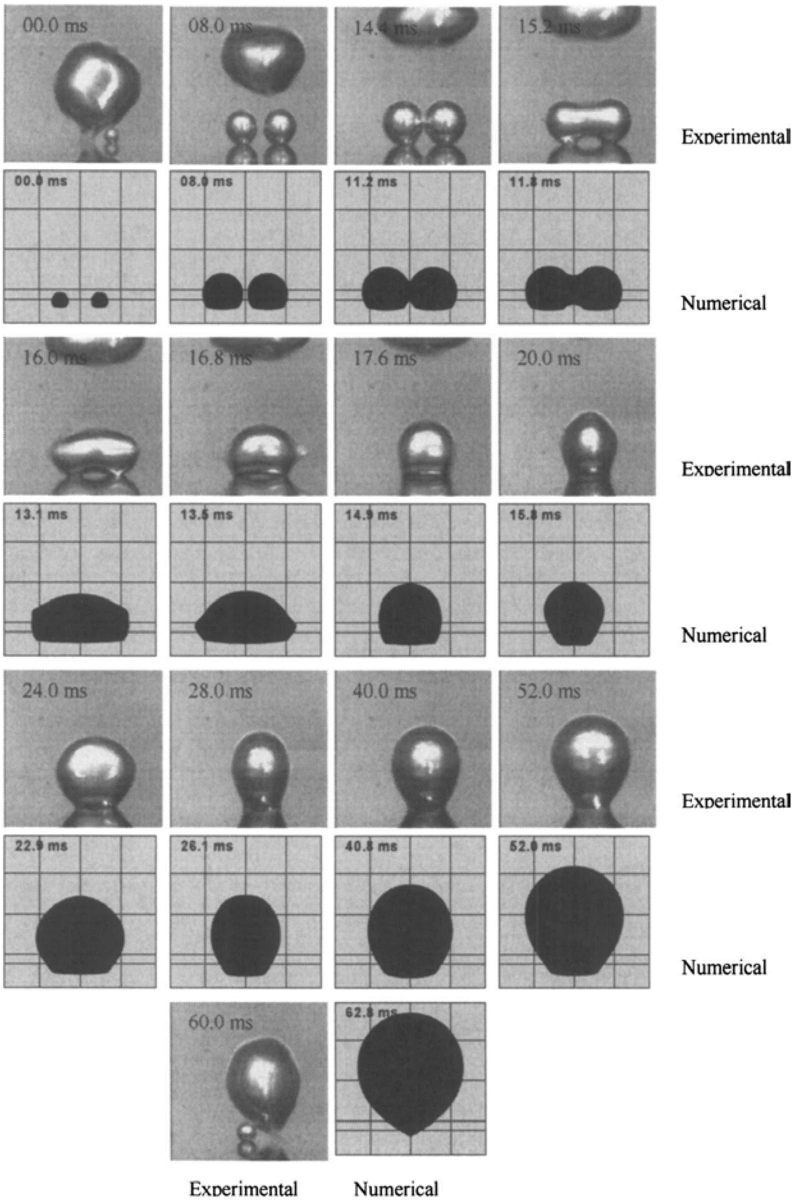


Fig. 2-1. Growth and Departure of Bubbles [2]

2.2. Call for Innovative System

The more enhanced heat transfer demands the more efficiency in the process of design nuclear plants. Not only economics but also sustainability, safety and reliability are major design considerations. And they are directly linked with the whole designed structure of nuclear plants. Moreover, the design is taken into consideration in which the prevention of nuclear power plant accidents. It requires more attentive accuracy.

So, the enhanced heat transfer is required to design nuclear plants more efficiently and to operate nuclear plants more economically. It will permit new designed nuclear plants and conventional nuclear plants to meet the Nuclear Energy System goals of sustainability, safety, reliability, and economics.

3. HEAT TRANSFER ENHANCEMENT

3.1. Major Concept of Heat Transfer Enhancement

Heat transfer is a discipline of thermal engineering that concerns the generation, use, conversion, and exchange of thermal energy and heat between physical systems. As such, heat transfer is involved in almost every sector of the economy. Heat transfer is classified into various mechanisms, such as thermal conduction, thermal convection, thermal radiation, and transfer of energy by phase changes.

The transfer of mass of differing chemical species, either cold or hot achieves heat transfer. While these mechanisms have distinct characteristics, they often occur simultaneously in the same system.

Heat conduction, also called diffusion, is the direct microscopic exchange of kinetic energy of particles through the boundary between two systems. When an object is at a different temperature from another body or its surroundings, heat flows in the direction so that the body and the surroundings reach the same temperature. When the same temperature is obtained, these bodies are in thermal equilibrium. Such spontaneous heat transfer always occurs from a region of high temperature to another region of lower temperature, as described by the second law of thermodynamics.

Convection is the concerted, collective movement of groups or aggregates of molecules within fluids (e.g., liquids, gases) and rheids, either through advection or through diffusion or as a combination of both of them. Convection of mass cannot take place in solids, since neither bulk current flows nor significant diffusion can take place in solids. Diffusion of heat can take place in solids, but that is called heat conduction. Convection can be demonstrated by placing a heat source.

Convective heat transfer is a mechanism of heat transfer occurring because of bulk motion (observable movement) of fluids. Heat is the entity of interest being carried and dispersed. It can be contrasted with conductive heat transfer, which is the transfer of energy by vibrations at a molecular level through a solid or fluid, and radiative heat transfer, the transfer of energy through electromagnetic waves.

By transferring matter, energy—including thermal energy—is moved by the physical transfer of a hot or cold object from one place to another. A practical example is thermal hydraulics. This can be described by the formula

$$Q = \upsilon \cdot \rho \cdot c_p \cdot \Delta T \qquad \textbf{(3-1)}$$

Where

Q is heat flux (W/m^2), ρ is density (kg/m^3), C_p is heat capacity at constant pressure ($J/(kg \cdot K)$), ΔT is the change in temperature (K), v is velocity (m/s).

The rate of acceleration due to gravity can be reduced by artificial flow. So, in this paper, Bubble cutter system, which can cut (and/or push and break) bubbles, is bodied out to drag bubbles.

It also makes the surface wet condition of heated surface and improves heat transfer. At low driving temperatures, no boiling occurs and the heat transfer rate is controlled by the usual single-phase mechanisms. As the surface temperature is increased, local boiling occurs and vapor bubble nucleate grows into the surrounding cooler fluid, and collapses. This is sub-cooled nucleate boiling, and is a very efficient heat transfer mechanism. At high bubble generation rates, the bubbles begin to interfere and the heat flux no longer increases rapidly with surface temperature (this is the departure from nucleate boiling, or DNB). At higher temperatures still, a maximum in the heat flux is reached (the Critical Heat Flux). The regime of falling heat transfer that follows is not easy to study, but is believed to be characterized by alternate periods of nucleate and film boiling. Nucleate boiling slows the heat transfer due to gas bubbles on the heater's surface; as mentioned, gas-phase thermal conductivity is much lower than liquid-phase thermal conductivity, so the outcome is a kind of "gas thermal barrier".

At higher temperatures still, the hydrodynamically-quieter regime of film boiling is reached. Heat fluxes across the stable vapor layers are low, but rise slowly with temperature. Any contact between fluid and the surface may probably lead to the extremely rapid nucleation of a fresh vapor layer. Finally, it causes a melting accident of the surface. So the extremely rapid nucleation of a fresh vapor layer must be prevented, critical heat flux is required in designing nuclear power plants.

Especially, motion of cool liquid is shown in (a) case of Figure 3-1, is worthy of notice. Bubble cutter system is a structure such as rotatable cutters (or movable cutters) or an artificial flow of fluid from the system, which can accelerate the motion of cool liquid or departure of bubbles.

In (a) case, black triangle section and red triangle section mean an artificial flow of fluid. When velocity from black triangle section to red triangle is increased by movement of an artificial flow of fluid, heat flux will increase.

More than two triangles can be intervened which is as shown in Figure 3-2. If the area between two triangle sections is small, the number of triangles passing over a surface of heat can be frequent. This short-term cut can be mated with the short formation cycle of bubbles.

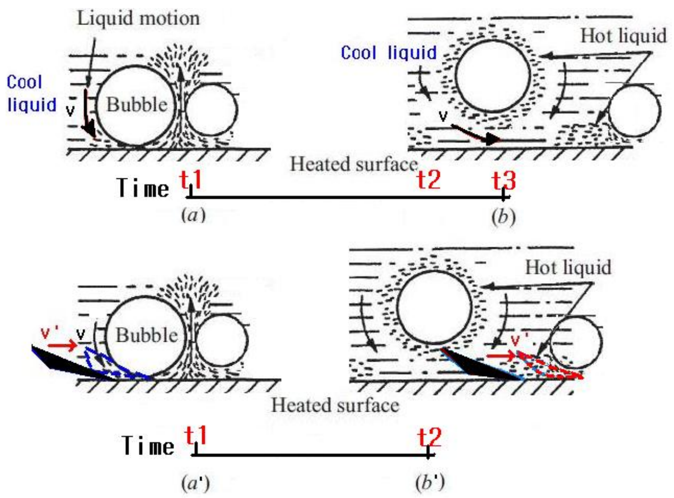


Fig. 3-1. A basic Concept of Proposed Bubble Cutter System

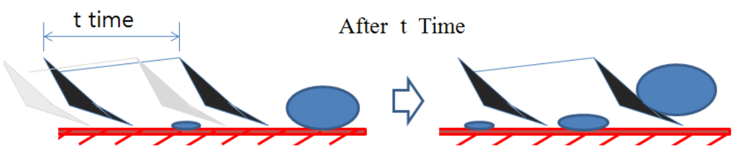


Fig. 3-2. A basic Concept of Proposed Cutter System

3.2. Proposed Methodology of Heat Transfer Enhancement

Artificial flow is not a discrete object. When the concept of bubbler cutter concludes a cutter as one of application methodologies, all application methodologies sometimes overlap and interweave. The artificial flow for heat transfer enhancement can be result from a shape, which is designed using shape design method.

3.2.1. Artificial flow As the Bubble Cutter

An artificial flow of fluid toward a heated surface can be produced by many sources such as by an impeller and a cold liquid induction pipe, which is shown in Figure 3-3. In case of Figure 3-3 (b), high pressure cold liquids is directly injected from outlets of the induction meanss such as pipes. And it pushes hot liquid by the action of artificial flow of fluid. Effective outlet velocity and flow rates and energy difference between hot liquid and the cold liquid will be only as strong as its enhanced critical heat flux.

When we consider the effect of artificial flow, the heat flux can be explained by using the Equation (3-1).

In reference, the ideal hydraulic power to drive a pump depends on the mass flow rate, the liquid density and the differential height. Of course, friction resistance in pipe also is under consideration. It is the static lift from one height to another, or the friction head loss component of the system is. Namely, the artificial flow needs energy consumption, it need a characteristic of “Overunity”. But, an energy-saving local artificial flow can be utilized if the intended cooling area is partly. The additional pumping power will be partially required to the occasion. Even though there may be frictional resistance in each channel of induced pipes, the required flow capacity is less than the case of pumping coolants to all areas.

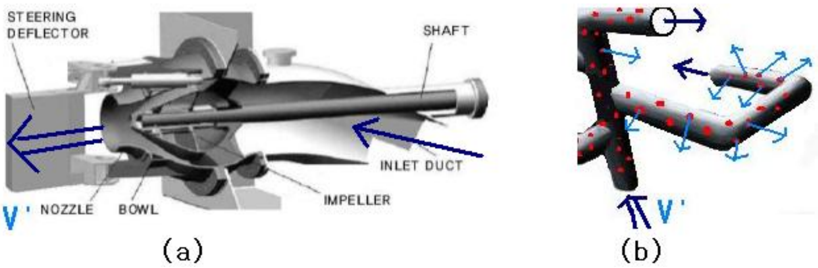


Fig. 3-3. Two Sample Sources of an Artificial Flow of Fluid

3.2.2. Application Prospect

Triangle sections mean a structure such as an artificial flow of fluid. When a pump pushes in a fluid to the targeted area, it needs pumping power. If the induced pipe which is connected with pump and other targeted area of cooling is so long, the system may need more high pumping power.

The artificial flow of fluid from nozzles can be designed for reciprocative motion, which is the way that resembles windshield wipers besides the energy-saving local artificial flow. If the heat source has a plate type surface, the artificial flow of fluid from nozzles can be designed for a to-and–fro motion. In that case, the forced flow requires a commitment of additional energy for pumping power.

But, shape design method can reduce the required power for the artificial flow of fluid. For example, in fluid dynamics, Bernoulli's principle states that for an inviscid flow, an increase in the speed of the fluid occurs simultaneously with a decrease in pressure or a decrease in the fluid's potential energy.

Bernoulli's principle can be applied to various types of fluid flow which includes a gas, and when we consider the effect of Bernoulli's principle in designing nuclear plants, the changed heat flux resulted from the increased speed of the fluid.

In the view of the thermal conductivity, we can consider a gas with temperature gradient in cooling pipes and tubes when a coolant is a gas. In such a case, the shape of cooling pipe can be modified for heat transfer enhancement.

For an example, if one side of tube is hot and the other side of tube cold as like as the cooling tubes did in fusion reactor, the heat flow is proportional to the number of particles which met hot side of the cooling tubes that cross the area (*A)* near hot side per second. This number is proportional to the product nvA where n is the particle density and v the mean particle velocity. The magnitude of the heat flow will also be proportional to amount of energy transported per particle so with the heat capacity per particle c and some characteristic temperature difference ΔT between hot particle and cool particle.

When we design to increase an inviscid flow relying on Bernoulli's principle state, an increase in the speed of the fluid occurs simultaneously with an increase in number of particles near hot aspect of cooling pipe. This is the shape design method which may be a useful design tool to a more localized strong cooling design.

In the important consideration in nuclear plants design, the number of particles or molecules near host aspect of cooling area per unit time must be increased. In that case, the needed number of particles or molecules can be obtainable if we increase the speed of fluid near the targeted cooling area, especially when it comes to liquid coolants.

In the some papers, the effects of a swirl tube have been discussed. From this point of view, the number of molecules of liquid near host aspect of cooling pipe per unit time increases due to the swirling movement. It comes to be the major reasons for the heat transfer enhancement.

4. MODELING OF FORCED ARTIFICIAL FLOW

4.1. Modeling of Bubble Behavior

In this study, the volume-of-fluid (VOF) model in a commercial CFD code ANSYS FLUENT 12.1 is used.

Original mesh was composed with 12545 nodes, 64 2D heating wall faces, zone, 64 2D pressure-outlet faces, 84 2D symmetry faces, 24320 2D interior faces, zone, 12288 quadrilateral cells.

Domain Extents:

x-coordinate: min (m) = 0.000000e+000, max (m) = 3.890000e-002

y-coordinate: min (m) = 0.000000e+000, max (m) = 1.168000e-001

In order to simulate the bubble departure in CFD code, the modeling of the bubble departure is performed.

Even though Previous SNU research such as the thesis (2010) of Sung Su Jun used self developed User Defined Function (UDF) in FLUENT code, this paper focuses on only the effects of the forced flow toward the departure of bubble [4].

So, its effects need to be analyzed by comparing with the general model of departure of bubbles. When the characteristic of the departure of bubble is investigated using the VOF model, the best known User Defined Function (UDF) in a tutorial -VOF is used, which is published by FLUENT Corporation [5].

While the temperature varies according to the distance from the heating boundary in the model in the tutorial, the temperature doesn’t vary according to the distance in this study.

Because the motivation of this work is to understand the effect of different type of artificial flow which can increase heat transfer, it makes sense to use the reliable model from ANSYS Corporation in order to detect the relation between analyzed data and inputted artificial flow.

There are 13 numbers of cases which are seen in Figure 4-1 and Figure 4-2.

The major understanding of the bubble behavior is the two simple trends in Figure 4-3. One is the growth of volume fraction continuously and the other is the sharp decrease of surface Nusselt Number whenever a bubbler departs from the film on heat.

Bubble departure may be related with the aspect of growth of the film and the aspect of unbalanced thermal circulation and its flow, which is seen in Figure 4-4.

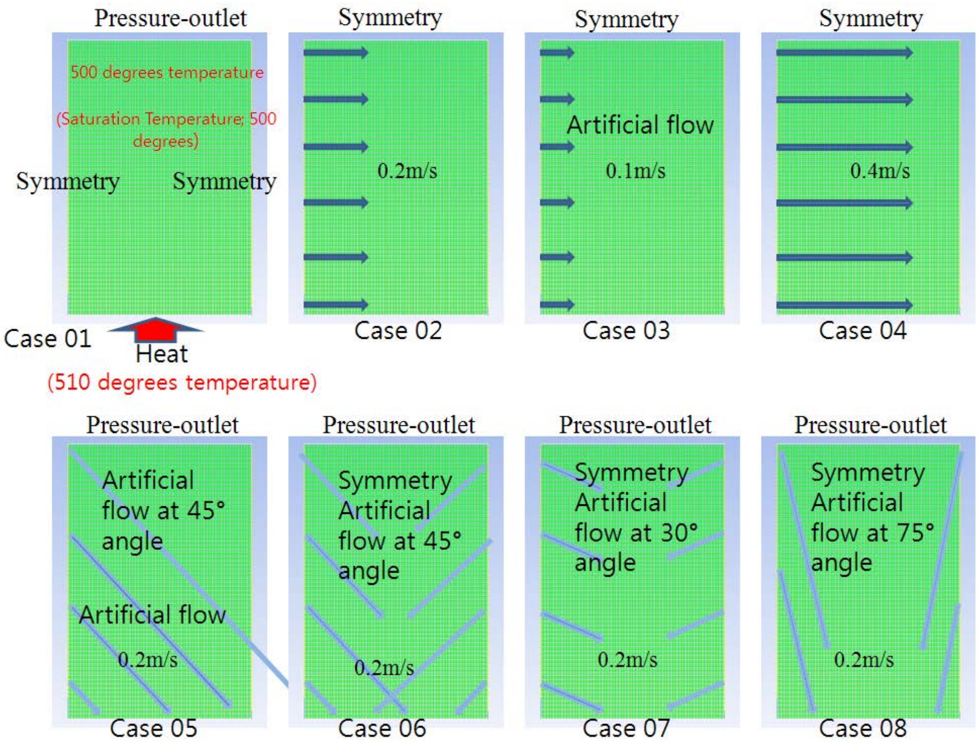


Fig. 4-1. Case (1-8)

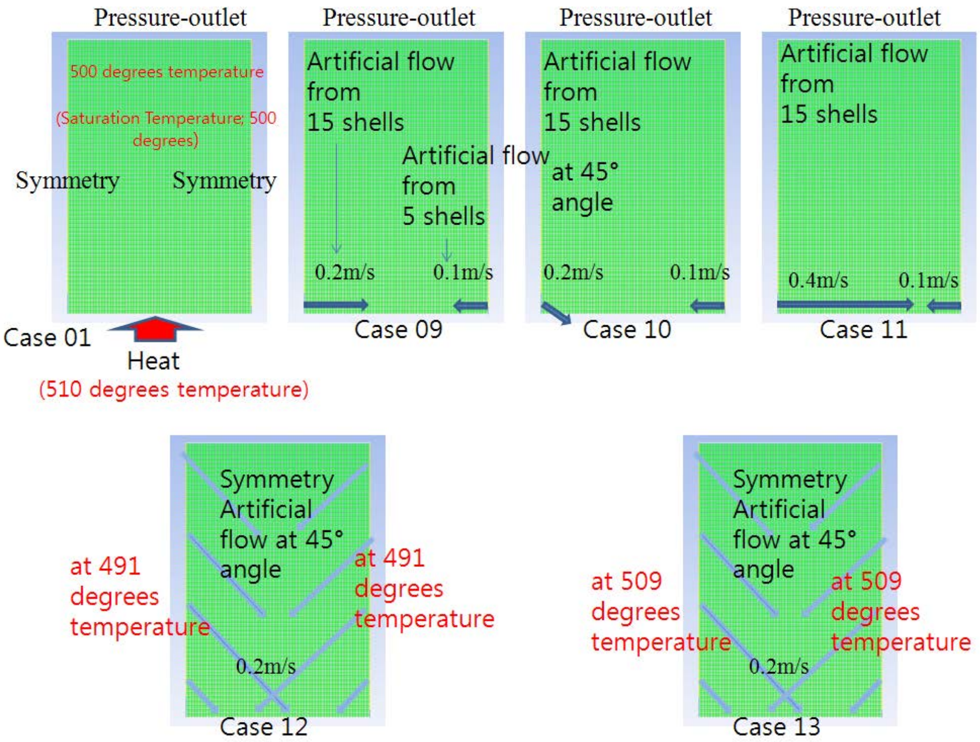
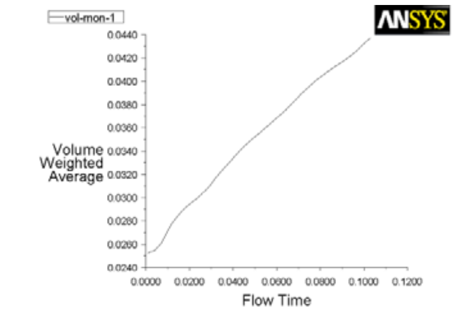
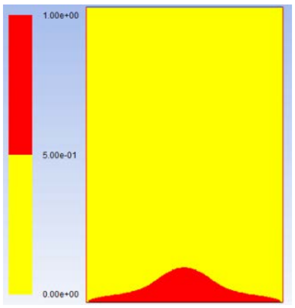
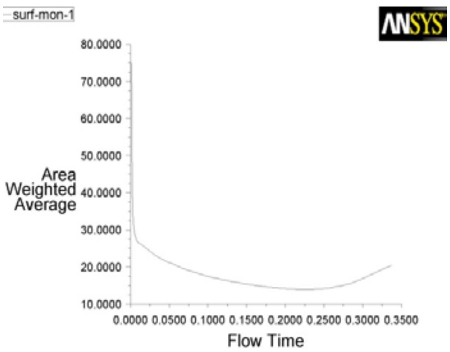
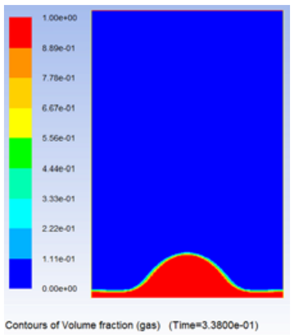
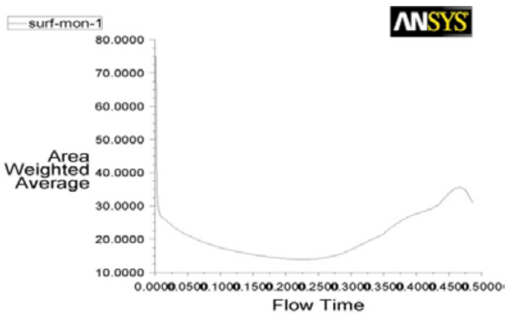


Fig. 4-2. Case (9-13)

<table>
<tr><th>No</th><th>Characteristic of Analyzed Date</th><th>Simulation</th></tr>
<tr><td>1</td><td>

Stage 1: Growth of volume fraction on heat and fluid</td><td>
</td></tr>
<tr><td>2</td><td>

Stage2: Increase of Surface Nusselt Number
(Pre-stage for Departure of a Bubble from heat)</td><td>
</td></tr>
<tr><td>3</td><td>

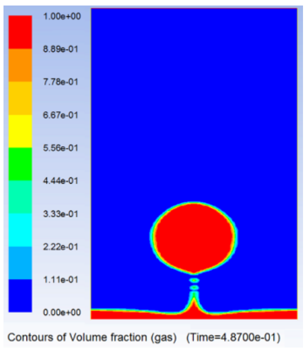
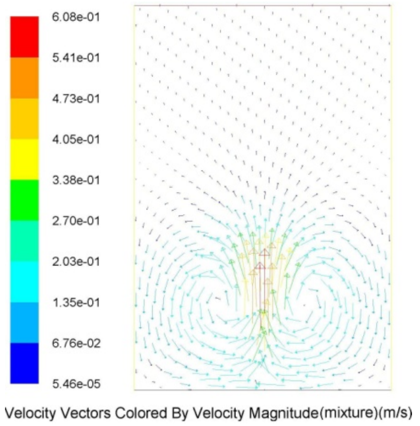
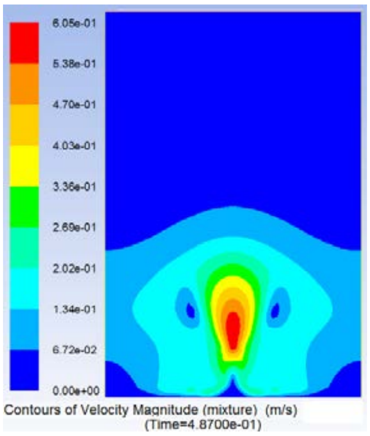
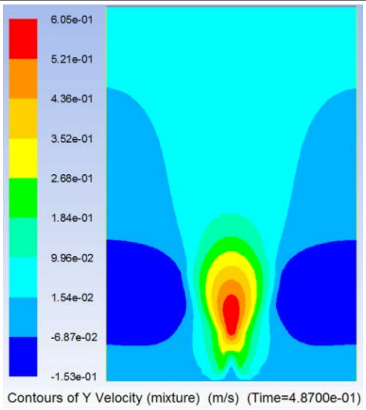
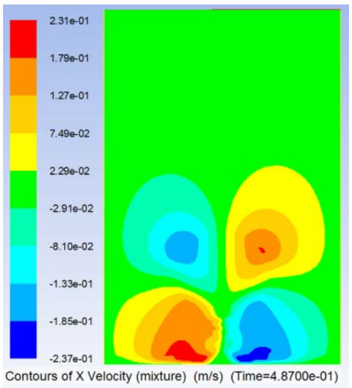
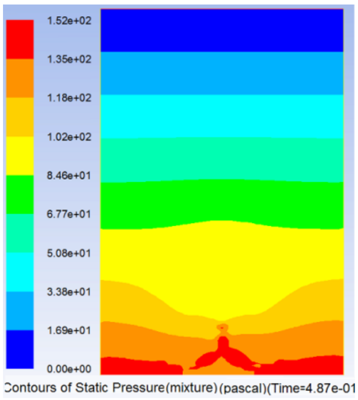
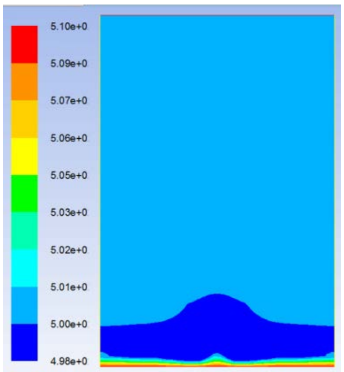
Stage3: Increase of Surface Nusselt Number
(Departure of a Bubble from heat)</td><td>
</td></tr>
</table>

Fig. 4-3. Characteristic of Basic Situation: Case 01

<table>
<tr><th>No</th><th colspan="2">Simulation</th></tr>
<tr><td>1</td><td>
</td><td>
</td></tr>
<tr><td>2</td><td>
</td><td>
</td></tr>
<tr><td>3</td><td>
</td><td>

Temperature</td></tr>
</table>

Fig. 4-4. Characteristic of Third Stage: Case 01

4.2. Modeling of Forced Artificial Flow

Because the direction, the temperature of flow is defined by inputted boundary condition, Figure 4-1 and Figure 4-2 show the inputted boundary condition of present modeling of the bubble behavior.

The characteristic of the present modeling of the bubble behavior are as follow.

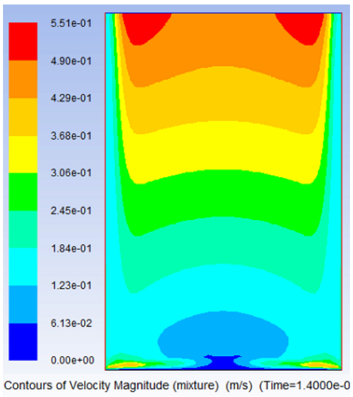
An artificial flow hits a film bubble, which is seen at the bottom and second photo in Figure 4-5, the film moves from the surface of heating boundary to out of cell, which was seen at the bottom and second photo in Figure 4-6. Therefore, in this study, additional symmetry flow is used to contain the bubble on the surface of heating boundary, which was seen at the first photo in Figure 4-7.

When asymmetry artificial flow was considered as an important case, this study discovered that that case can’t show the bubble departure. As the number of symmetry artificial flow cases increased, so has the number of cases.

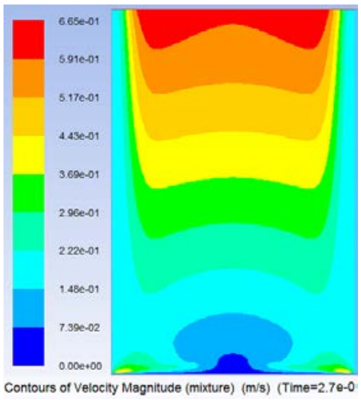
For example, 0.2m/s symmetry artificial flow at 45° angle, 30° angle and 75° angle are the added cases. 0.2m/s semi-symmetry artificial flow, 0.4m/s semi-symmetry artificial flow and etc are the another cases. In Case 09, Case 10 and Case 11, the forced flow from 15 right shells is designed to determine whether the energy-saving local artificial flow is effective.

This behavior demonstrates the effects of asymmetry artificial flow is to wash over heat and sweep bubbles from heat to near boundary. But the development patterns of a bubble departure in the symmetry artificial flow cases are similar to the case in the tutorial -VOF [5].

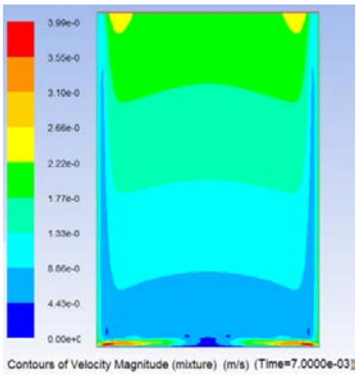
The symmetry artificial flow cases also will be applied to the different temperature artificial cases.



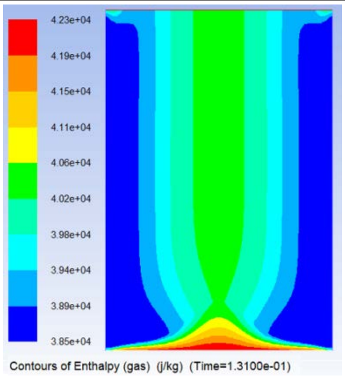
Case 05 ; 0.2m/s symmetry artificial flow at 45° angle



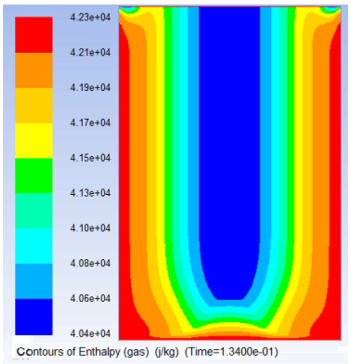
Case 07 ; 0.2m/s symmetry artificial flow at 30° angle



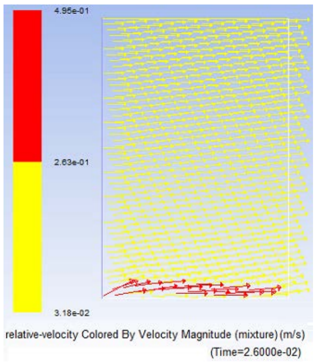
Case 08 ; 0.2m/s symmetry artificial flow at 75° angle



Case 12 ; 0.2m/s symmetry artificial flow at 491 degrees temperature



Case 13 ; 0.2m/s symmetry artificial flow at 491 degrees temperature



Additional Case ; 0.2m/s asymmetry artificial flow from the lowest 15 faces

Fig. 4-5. Cases of Artificial Flow (1)

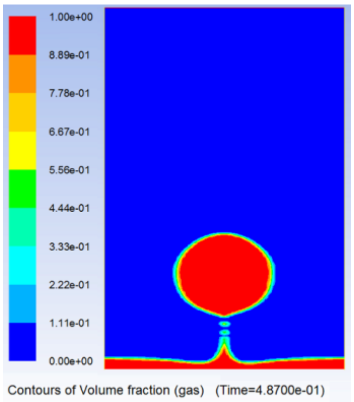
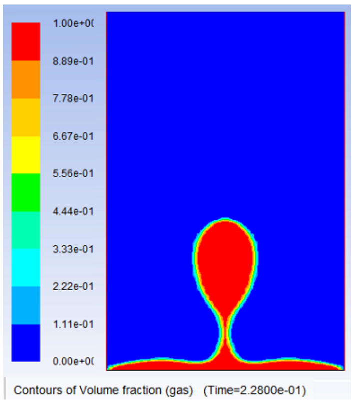
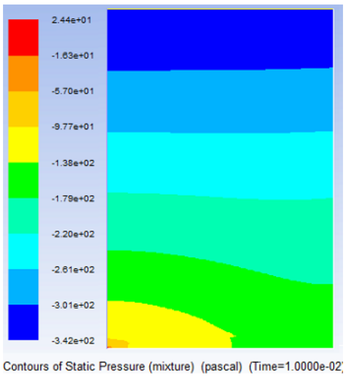
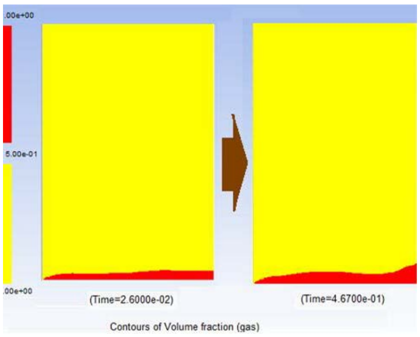
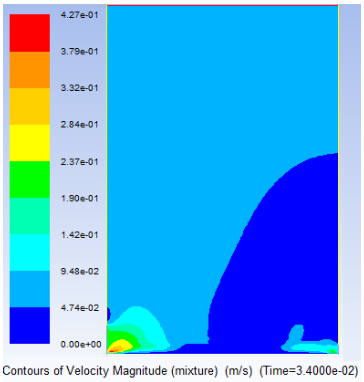
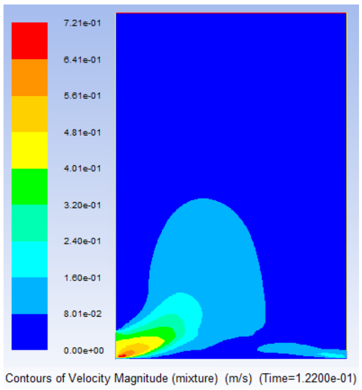
<table>
<tr><td></td><td colspan="2">Simulation</td></tr>
<tr><td rowspan="2">1</td><td>
</td><td>
</td></tr>
<tr><td>Case 01 ; Original case
(Round shape of early bubble)</td><td>Case 05 ; Symmetry artificial flow at 45° angle
(Shape of bubble, speed of Bubble departure)</td></tr>
<tr><td rowspan="2">2</td><td>
</td><td>
</td></tr>
<tr><td>Additional Case; 0.2m/s asymmetry artificial flow from the lowest 15 faces
(Unbalanced distribution of Pressure)</td><td>Case 02, 03, 04, 05, additional case; 0.2m/s asymmetry artificial flow from the lowest 15 faces
(Shift of Bubble on surface)</td></tr>
</table>

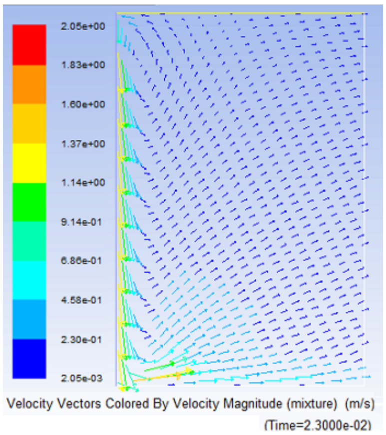
Fig. 4-6. Difference between Two Cases



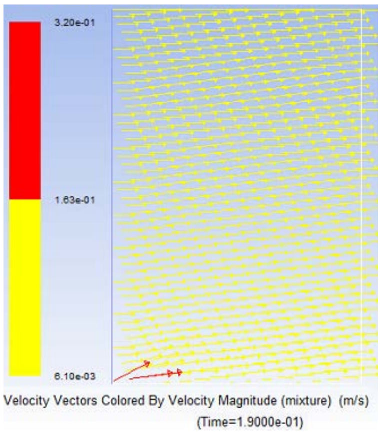
Case 09 ; 0.2m/s artificial flow from right lowest 15 faces Vs. 0.1m/s artificial flow from right lowest 5 faces



Case 11; 0.4m/s artificial flow from right lowest 15 faces Vs. 0.1m/s artificial flow from right lowest 5 faces



Case 05; 0.2m/s asymmetry artificial flow at 45° angle



Case 05 ; 0.1m/s asymmetry artificial flow from the lowest 15 shells

Fig. 4-7. Cases of Artificial Flow (2)

5. ANAYSIS OF EFFECTS OF FORCED ARTFICIAL FLOW

5.1. ANSYS VOF Modeling Result

5.1.1. Departure of Bubble

In this study, different types of artificial flow by comparing results with VOF modeling. The departure of bubble can be accelerated by the artificial flow. It means that the artificial flow can increase heat transfer.

Results for 0.2m/s symmetry artificial flow at 45° angle, 30° angle and 75° angle, Figure 5-1 shows development patterns of bubble departure. It seems that the different pressure to the bubbles can induce different bubble behavior. This behavior demonstrates that effects of an effective direction of artificial flow to a bubble for cutting off the bubbles. The more the forced flow hits bubble effectively, the more increased heat transfer is.

Results for 0.2m/s symmetry artificial flow at higher temperature (509 degrees) and lower temperature (491 degrees), which are seen in Figure 5-2, number 5, number 6, show the fact that the different temperature to the bubbles can induce different bubble behavior. This behavior demonstrates that effects of a low temperature artificial flow to a bubble increase these effects inducing sharper and more increased heat transfer.

5.1.2. Heat transfer

Since the Nusselt Number (Nu) is the ratio of convective to conductive heat transfer across a boundary, the surface Nusselt Number is the ratio of convective to conductive heat transfer of the heater. A larger Nusselt number corresponds to more active convention.

In the modeling of this study, the surface Nusselt Number is determined by the thickness of film bubble and the crashing flow of liquid. When a bubble departures from surface of heat, the surface Nusselt Number decreases.

So, it is observed that the increased speed of bubble departure can increase heat transfer. The simulation slows these qualitative features of the bubble departure.

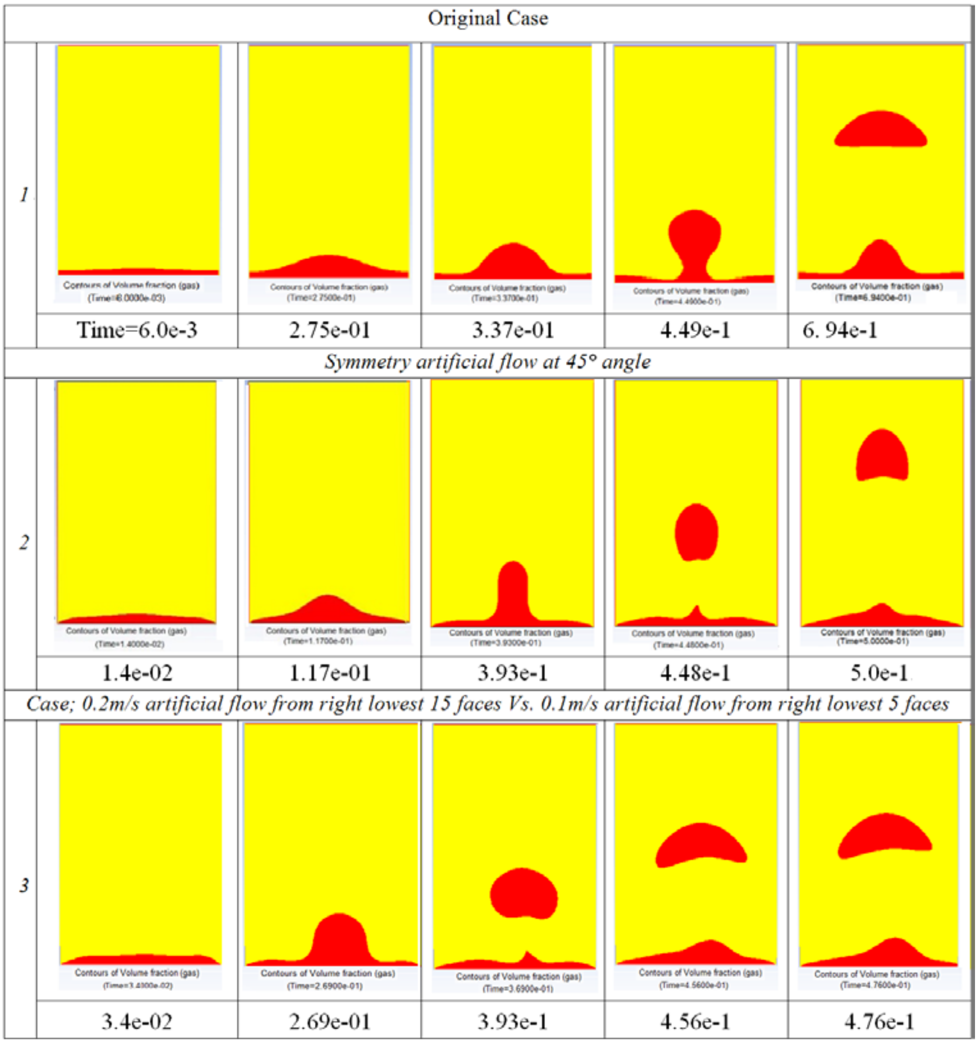


Fig. 5-1. Departure of Bubble (1)

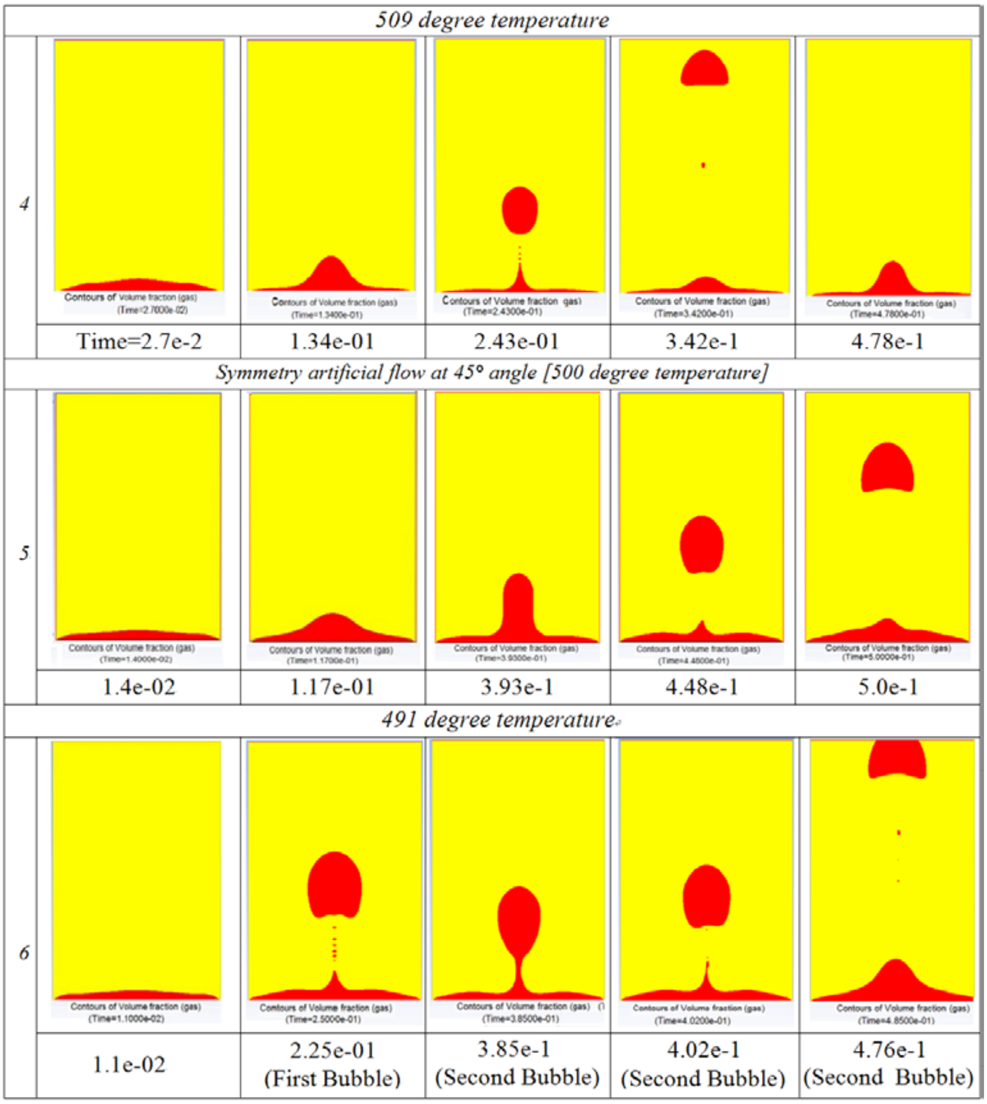


Fig. 5-2. Departure of Bubble (2)

5.2. Results and Discussion:

.

We now turn to a more challenging test of deductive reasoning to reach conclusion.

In past research, Damir Juric and Gretar Tryggvason conferred the shape of bubbles. They showed that a parent bubble assumes a skirted shape [6]. It can be assumed that the increased departure of a bubble by the symmetry inlet flow may help the film to transform from a film into a bubble shape. When the phase change process is deal with by adding a user defined function (UDF) source term in each phase, a model is used to consider the effect of surface tension [7]. So, not only shapes of bubbles but also its surface tension can be considered in explaining the departure of a bubble.

Asymmetry velocity makes the flow drag bubbles from an original place to another place. Consequently, any a bubble could not be formed in case 02, case 03, case 04 and case 05. So, the heat transfer enhancement was equal to velocity. The phenomenon is explained in Figure 5-3.

The phenomenon is explained in Figure 5-4. It shows easy growth and departure of bubbles on heat can increase heat transfer. While the artificial flow was became with a small quantity of liquid in case 09, the efficiency of formed liquid pressure in cutting for bubbles makes the artificial flow to compete on equal terms with the artificial flow in case 02. We make sure it meets the assumption that the more rapid rise in the number of bubble departure will increase the heat transfer.

A flow of warm liquid can’t increase heat transfer highly in case 13. The lesson from this study is the fact that the artificial flow should make it easier to become a bubble shape. In case 08, the artificial flow at 75° angle permits the center of bubble to be free of pressure. Consequently, the case 08 achieves improvement in heat transfer as compared with case 06 and case 07.

Generally, it is found that wall contact angle has a great influence on the morphology of bubble [8]. It seems that the pressure of liquid may help bubble to form an appropriable content angle in case 08.

Figure 5-6 shows several forms of bubble departure. Table 6-1 shows crude bubble departure frequency within 4.80e-1 second. In the case 10, the bubble departure frequencies are nearly the same except for the first bubble departure frequency.

While Surface Nusselt Number defines what heat transfer enhancement is alike, aspects of the bubble departure frequencies are often variously not to have power of explanation. For example, Surface Nusselt Number of case 03 has power of explanation about its heat transfer enhancement while its aspect of bubble departure is to be pushed out without any chance of bubble formation.

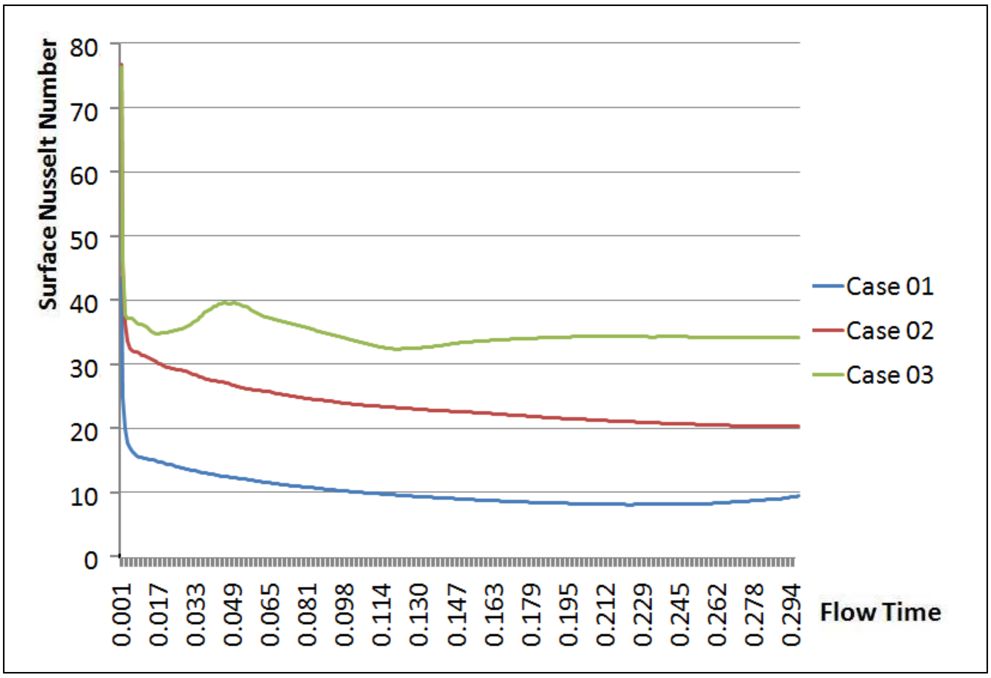


Fig. 5-3. Surface Nusselt Number on Heat (1)

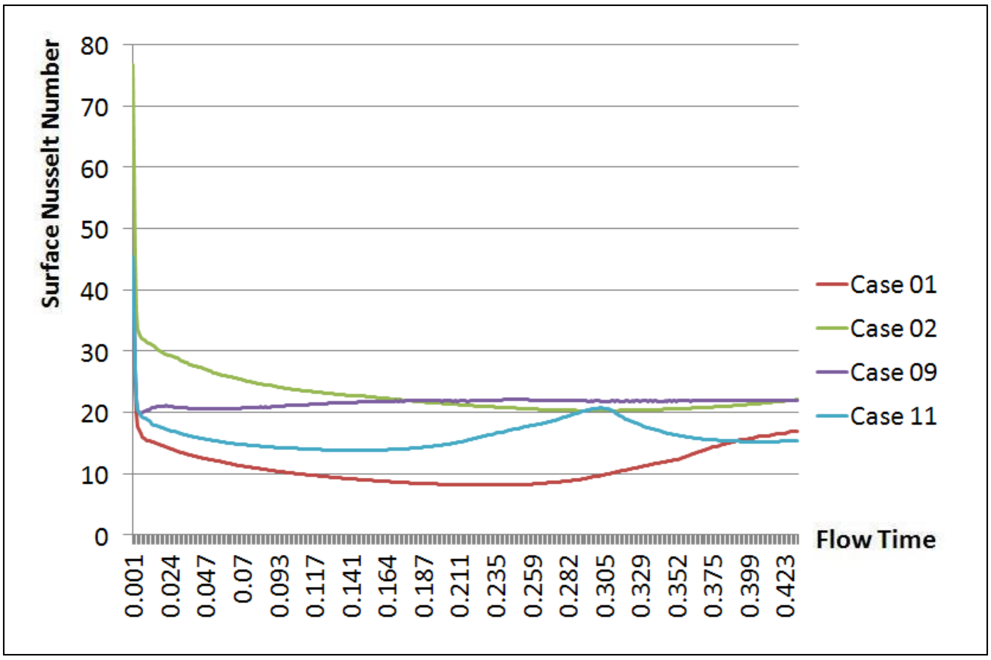


Fig. 5-4. Surface Nusselt Number on Heat (2)

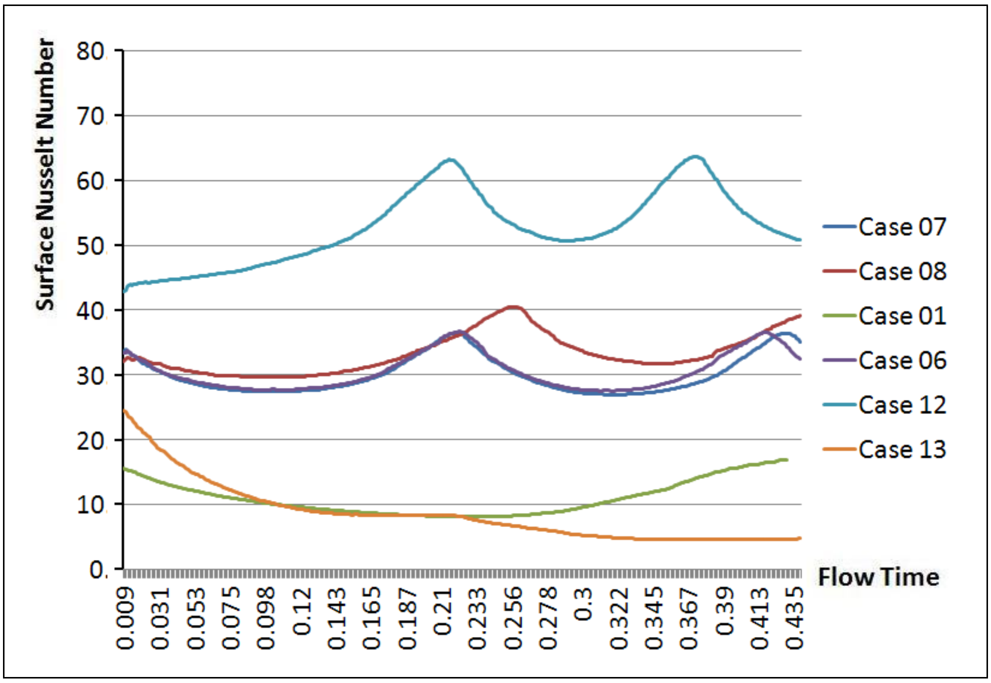


Fig. 5-5. Surface Nusselt Number on Heat (3)

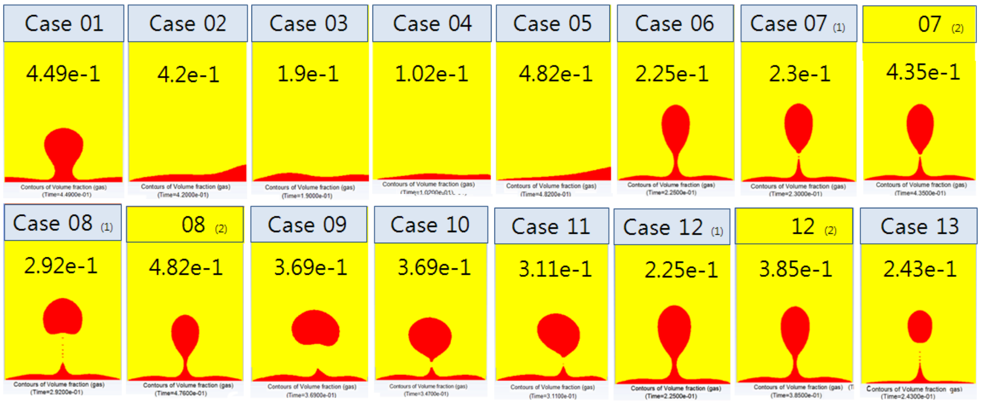
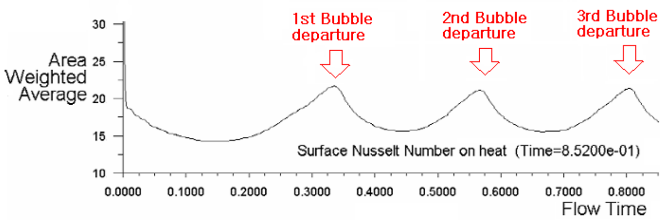


Fig. 5-6. Bubble Departure

Table 6-1. Bubble Departure Frequency

No.	First Bubble Departure		Second Bubble Departure within 4.80e-1	
1	Over than 4.80e-1	–	–	–
2	-	–	–	–
3	-	–	–	–
4	-	–	–	–
5	-	-	-	-
6	2.29 e-1	1st frequency; 2.29 e-1	4.35e-1	2nd frequency; 2.04 e-1
7	2.30 e-1	1st frequency; 2.30 e-1	4.35e-1	2nd frequency; 2.05 e-1
8	2.45e-1	1st frequency; 2.45 e-1	4.40e-1	2nd frequency; 1.95 e-1
9	3.51e-1	1st frequency; 3.51 e-1	-	-
10	3.47e-1	1st frequency; 3.5 e-1	- * Reference	- * Reference
11	3.11e-1	1st frequency; 3.11 e-1	-	-
12	2.29e-1	1st frequency; 2.29 e-1	3.87e-1	2nd frequency; 1.58 e-1
13	2.33e-1	1st frequency; 2.33 e-1	-	-

* Reference; Third bubble departure time is 5.76e-1. (3nd frequency; 1.45e-1) – Except for 1st frequency, the frequency may be similar to the next frequency). Fourth bubble departure time is 8.20e-1. (4th frequency; 1.44e-1)



6. Heat Transfer Design Applying the Proposed Methodology

6.1. Application Methodology

At first, an artificial flow of fluid toward a heated surface can be produced by many sources such as an impeller and a cool. Further, in case of Figure 3-1 (a), triangle sections means a structure such as the bubble cutter or an artificial flow of fluid. In the case of making a real cutter rotating on a heated surface in order to cut (and/or bush, break) bubbles, there can be two type of cutter mounting, which is shown in Figure 6-1.

The artificial flow and bubble cutter are not discrete objects, they sometimes overlap and interweave. For example, Bubble cutter can be not only a structure of cutter but also an artificial flow which is impelled toward bubble through induced pipes or impeller. Triangle sections mean a structure such as a monolithic bubble cutter or an artificial flow of fluid. In the case of a structure of making a real cutter rotating on a heated surface in order to cut (and/or bush, break) bubbles, it needs rotational power. If the system doesn't need high speed of rotation, it can be designed to rotate outer surface or inner surface of pipe by using nearby flow of fluid with wing.

In general, rotational energy will be proportional to flow power of a fluid. Generally, flow energy such as wind energy is the kinetic energy of fluid such as air in motion. Total flow energy flowing through an imaginary area A during the time t is concern with density of fluid and speed of fluid. The shape design which induces Bernoulli Effect can lessen the needed power.

When heating surface is a slab, the shape design can be properly taken into account. In such a case, we can design a bubble cutter for reciprocative motion besides the shape design method, which is the way that looks like windshield wipers.

Secondly, hybrid cooling design method means the combination of a certain cooling system. When a pump pushes in a fluid such as liquid or gas to the targeted area, it can push different kind of coolant to the targeted area. Of course, gas mixture or fluid mixture will come into being. Whereupon, when the different kinds of coolants circulate through the nuclear system or fusion reactor system, the mixture of gases or fluids needs to be separated the original coolant and the newly input coolant. These design concepts of hybrid cooling design method are necessary if some local areas need to be cooled by a coolant especially.

Major proposed methodologies are summarized in Table 6-1. It is essential not to spoil the safety of nuclear plants when it is involved in heat transfer enhancement for economics.

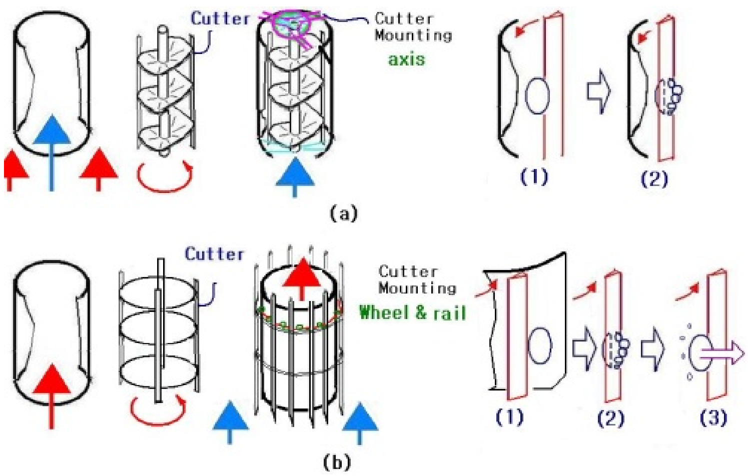


Fig. 6-1. Type of Cutter Mounting

Table 6-2. Proposed Methodologies

No.	Title	Coolant	Heat transfer / CHF
1	Bubbler cutter (structure type)	Liquid	O / O
2	Bubbler cutter (artificial flow type)	Liquid	O / O
3	Artificial flow (Impeller type)	Liquid, gas	O / O
5	Shape design	Liquid, gas	O / O
6	Hybrid cooling design	Liquid, gas (allowed for mixture)	O / O

6.2. Heat Transfer Design in Nuclear Power Plants

A nuclear power plant as a thermal power station has the heat source, a nuclear reactor as a primary system, and steam turbine as a secondary system. In Boiling Water Reactor (BWR), when heat is produced by fission in a nuclear reactor, it produces steam.

In Nuclear Power Plant (NPP), especially in a Pressurized Water Reactor (PWR), the primary coolant is pumped under high pressure to the reactor core where it is heated by the energy generated by the fission of atoms. The heated water then flows to a steam generator where it transfers its thermal energy to a secondary system where steam is generated and flows to turbines which, in turn, spin an electric generator. In contrast to a boiling water reactor, pressure in the primary coolant loop prevents the water from boiling within the reactor. A U-shaped heat exchanger is used to transfer the thermal energy to a secondary system. All Light Water Reactors (LWRs) use ordinary water as both coolant and neutron moderator.

A bubble cutter system may be not embodied easily due to an intricate piece of machinery of reactor core. But an artificial flow of water can be produced by the bubble cutter system which has an impeller for watering toward a heated surface of the reactor core. Of course, all heated water by each nuclear fuel rod of the reactor core will not be affected by an artificial flow of fluid in every part of reactor in this way.

In the case of a heat exchanger system, while some bubbles flow under the influence of an artificial flow of water, some bubbles are not seriously affected by produced swirls. If all bubbles are affected by a bubble cutter (and/or an artificial flow), each pipe needs to be surrounded by bubble cutters that was connected by a rotor. In such a case, the bubble cutter system cannot be applied for curved parts of the pipes of the heat exchanger. But, many induction pipes which are inserted between the pipes can be with luck on its side, a cool liquid induction pipe type artificial flow can be produced from the engineered fluid flow design.

The surface-condenser has many tubes. If the rotating cutter can get rid of non-condensable gases, the overall heat transfer coefficient based on outside tube area increases. Due to presence of many tubes, it may be impossible to rotate the cutters by a rotor or a pump.

The fact that the system needs a pumping power or rotating power may make the cutters as an impracticable proposal. If the cutters rotate tube relying on a stream of water that has an effective head drop, it drags insoluble gas out centrifugally. In order to utilize Bernoulli's principle, when we narrow a tube, the changed heat flux resulted from the increased speed of the fluid can be elicited by using the equation (3-1).

It also may be a useful design method to utilize Bernoulli's principle in the nuclear reactor design and steam generator design.

In the view of the thermal conductivity, even though it does not appeal, we can consider a gas as a coolant with temperature gradient in cooling pipes and tubes. To take a concrete instance, we can consider the example of fusion reactor. The heat flow is proportional to the number of particles which met hot side of the cooling tubes that cross the area near hot side per second. So, the thickness of the coolant tube needs to be reduced. This is shape design method which may be a useful design tool to a more localized strong cooling design.

7. CONCLUSION AND FUTURE WORK

The conventional implementation methodology of heat transfer enhancement focused on the flow boiling Critical Heat Flux (CHF) enhancement with surfactant solutions under atmospheric pressure. Surfactant solutions or surface conditions have an effect on the behavior of occurrence bubbles on a heated surface [9][10].

This paper proposes an artificial flow design method and cutters design method and shape design method for another possible improvement in heat transfer enhancement. The implementation methodology was studied to depict some types of bubble breaking (and/or pushing, breaking) system or an artificial flow of fluid that can push hot liquid and bubbles on a surface of heater [11].

In condenser, film on the surface of tubes is made of non-condensable gas out. Even though the proposed artificial flow method can drag film, the enhanced heat transfer results from the increased heat flux.

The heat transfer designs have to be carried out with their efficiencies approaching the ideal value in order to achieve the maximum overall system effective applying proposed methodologies [12][13][14][15][16].

A bubble cutter system may not be embodied easily due to an intricate piece of machinery of reactor core in Pressurized Water Reactor (PWR) and all Light Water Reactors (LWRs). But, in the case of a surface-condenser, the strong forced flow of liquid which can be induced by a proper shape design can increase overall heat transfer coefficient. Even in the case of a fission reactor, we can narrow a tube to utilize Bernoulli's principle by adjusting the depth of the tube. At that time, the changed heat flux resulted from the increased speed of the fluid.

We researched the characteristics of bubbles departure which has an effect on the heat transfer characteristics of the nuclear system. After understanding the increased rate of bubble departure, it focused on heat transfer enhancement using artificial flow, which can develop into the concept of bubble cutter.

At first, bubble cutter systems can be consisted of an artificial flow.

In this study, when we simulate the bubble departure in CFD code, the modeling of the bubble departure is performed using the well-known Volume of Fluid (VOF) model. It is found that the characteristic of the bubble departure is significantly different from that of the bubble in every aspect including speed of bubble departure, the size of bubble, the velocity of bubble movement, moving trajectory and coalescence of bubbles.

Secondly, the shape design method can induce more increased flow. Hybrid cooling design method for another possible improvement in heat transfer enhancement can be utilized using Bernoulli's principle from the shape. The changed heat flux resulted from the increased speed of the fluid can be elicited by narrowing a tube to utilize Bernoulli's principle. And it also can be elicited by inputting different kind of high specific heat coolant to the targeted area considering its affect to total heat flux.

In this study, we researched augmenting bubble behavior on heat with external flow using volume of fluid model. The subject was also concerned about the implementation methodology of heat transfer enhancement in nuclear power plants

To investigate meaning of enhanced heat transfer, we find that a basic meaning of proposed cutters design method, artificial flow design method has a difficulty to be realized in nuclear plants design. For example, while the artificial flow of fluid can lessen the effect of non-condensable gases centrifugally on surfaces of tubes of a condenser using forced artificial flow, we cannot insert inducing pipes to the condenser. Therefore shape design which utilizes the Bernoulli's principle can be considered.

In all cases, computational fluid dynamic analysis may be needed to analyze its effect exhaustively.

REFERENCES

1. Ho Seon Ahn and Moo Hwan Kim, “The Boiling Phenomenon of Alumina Nanofluid near Critical Heat Flux”, International Journal of Heat and Mass Transfer, Vol. 62, PP 718-728 (2013).
2. A. Mukherjee and V.K. Dhir, “Study of Lateral Merger of Vapor Bubbles During Nucleate Pool Boiling”, Journal of Heat Transfer, Vol. 126, PP. 1023-1039, USA (2004).
3. Tomqaki Kungi, “Brief Review of Latest Direct Numerical Simulation on Pool on Pooland and Film Boiling”, Nuclear Engineering and Technology 218, Vol. 44, PP. 847-854, Korea (2012).
4. Sung Su Jun, “Numerical Study of Condensing Bubble in Subcooled Boiling Flow Using Volume of Fluid Model”, Seoul National University MS Dissertation, Korea (2010).
5. Fluent. Inc. , “Tutorial: Horizontal Film Boiling”, U.S.A. (2002).
6. Damir Juric and Gretar Tryggvason, “Computations of Boiling Flows”, Int. J. Multiphase Flow, Vol.24, No. 3, PP. 387-410, U.K. (1998)
7. ZR Lin, SF Wang, R Shirkashi, and LW Zhang, “Simulation of a Miniature Oscillating Heat Pipe in Bottom Heating Mode Using CFD with Unsteady Modeling”, International Journal of Heat and Mass Transfer, Vol. 57, PP. 642-656, UK (2013)
8. Lei Guo, Shu-Sheng Zang, Lin Cheng, “Study on Characteristics of Vapor-Liquid of Vapor-Liquid Two-Phase Flow in Mini-Channels”, Nuclear Engineering and design, Vol. 241, PP.4158-4164, U.S.A., (2011)
9. Yong Hoon Jeong, Mohammad Sohail Sarwar and Soon Heung Chang, “High Current Regulated Flow Boiling CHF Enhancement with Surfactant Solutions under Atmospheric Pressure”, International Journal of Heat and Mass Transfer, 25 Sep. UK (2007).

10. Ho Seon Ahn, Chan Lee, Hyungdae Kim, HangJin Jo, SoonHo Kang, Joonwon Kim, Jeongseob Shin and Moo Hwan Kim, "Pool boiling CHF enhancement by micro/nanoscale modification of zircaloy-4 surface", Nuclear Engineering and Design, Vol. 240, PP. 3350-3360, UK (2010).
11. Chan Hee Jung and Kune Y. Suh, "Enhanced CHF with Bubble Cutter and Artificial Flow in Nuclear Power Plants", Transactions of the Korean Nuclear Society Autumn Meeting, Gyeongju, October 24-25, Korea (2013).
12. Nakaharai Hiroyuki, Takami Satoshi, Yokomine Takehiko, Ebara Shinji and Shimizu Akihiko, "Numerical Study of Heat Transfer Characteristics in a Tube with Regularly Spaced Twisted Tape", Fusion Science and technology, Vol.52, PP. 855-859, UK, (2007).
13. Novog. D. R., Yin S. T. and Chang J. S., "Recent Advances in High Heat Flux Smooth and Swirl Flow Boiling of Water", Fusion Science and Technology, Vol. 52, PP. 880-884, UK (2007).
14. Aleksey V. Dedov, Alexander T. Komov, Alexander N. Varava, Victor V. Yagov, "Hydordynamics and Heat Transfer in Swirl Flow under Condition of One-Side Heating. Part 2: Boiling Heat Transfer. Critical Heat Fluxes", International Journal of Heat and Mass Transfer, Vol.53, PP. 4966-4975, UK (2010).
15. Raj M. Manglik, "Swirl Flow Heat Transfer and Pressure Drop with Twisted-Tape Inserts", Advances in Heat Transfer, Vol.36, PP. 183-266, UK (2010)
16. X. Fu, P. Zhang, C.J. Huang, R.Z. Wang, "Bubble Growth, Departure and the Following Flow Pattern Evolution during Flow Boiling in a Mini-Tube", International Journal of Heat and Mass Transfer, Vol.53, PP. 4819-4831, UK (2010)

17. APPENDIX

Source of User Define Function reference; [5]

```
#include "udf.h"
#include "sg.h"
#include "sg_mphase.h"
#include "flow.h"
#include "mem.h"
/***************************************************************/
/* UDF for specifying an interfacail area density                */
/***************************************************************/
DEFINE_ADJUST(area_density, domain)
{
  Thread *t;
  Thread **pt;
  cell_t c;
  Domain *pDomain = DOMAIN_SUB_DOMAIN(domain,P_PHASE);
  real voidx, voidy, voidz=0;
  {
  Alloc_Storage_Vars(pDomain,SV_VOF_RG,SV_VOF_G,SV_NULL);
  Scalar_Reconstruction(pDomain, SV_VOF,-1,SV_VOF_RG,NULL);
  Scalar_Derivatives(pDomain,SV_VOF,-1,SV_VOF_G,SV_VOF_RG,
                                    Vof_Deriv_Accumulate);
   }
   {
      Alloc_Storage_Vars(domain, SV_T_RG, SV_T_G,   SV_NULL);
      T_derivatives(domain);
      Free_Storage_Vars(domain, SV_T_RG, SV_NULL);
```

```
        }
          mp_thread_loop_c (t,domain,pt)
                  if (FLUID_THREAD_P(t))
                     {
                        Thread *tp = pt[P_PHASE];
                        begin_c_loop (c,t)
                           {
#if RP_3D
                  C_UDMI(c,t,0) = (C_VOF_G(c,tp)[0]*C_T_G(c,t)[0]+
                  C_VOF_G(c,tp)[1]*C_T_G(c,t)[1]+C_VOF_G(c,tp)[2]*C_T_G(c,t)[2]);
#endif
#if RP_2D
                  C_UDMI(c,t,0) = (C_VOF_G(c,tp)[0]*C_T_G(c,t)[0]+
                  C_VOF_G(c,tp)[1]*C_T_G(c,t)[1]);
#endif
                           }
                end_c_loop (c,t)
               }

  Free_Storage_Vars(pDomain,SV_VOF_RG,SV_VOF_G,SV_NULL);
  Free_Storage_Vars(domain, SV_T_G, SV_NULL);
}
DEFINE_SOURCE(gas, cell, thread, dS, eqn)
{
   real x[ND_ND];
   real source;
   Thread *tm = THREAD_SUPER_THREAD(thread);
   Thread **pt = THREAD_SUB_THREADS(tm);
   real Kl = C_K_L(cell, pt[1])*C_VOF(cell, pt[1]),
   Kg = C_K_L(cell, pt[0])*C_VOF(cell, pt[0]);
```

```
    real L = 1e5;
    source = (Kl+Kg)*C_UDMI(cell,tm,0) / L;
    C_UDMI(cell, tm, 1) = source;
    C_UDMI(cell, tm, 2) = -source*L;
    dS[eqn] =0;
    return source;
}
DEFINE_SOURCE(liquid, cell, thread, dS, eqn)
{
    real x[ND_ND];
    real source;
    Thread *tm = THREAD_SUPER_THREAD(thread);
    Thread **pt = THREAD_SUB_THREADS(tm);
    source = -C_UDMI(cell, tm, 1);
    dS[eqn] = 0;
    return source;
}
DEFINE_SOURCE(energy, cell, thread, dS, eqn)
{
    real x[ND_ND];
    real source;
    Thread *tm = thread;
    source = C_UDMI(cell, tm, 2);
    dS[eqn] = 0;
    return source;
}
/*********************************************************************/
/* UDF for initializing flow field variables                         */
/*********************************************************************/
DEFINE_INIT(my_init_function, domain)
```

```
{
 Thread *t;
 Thread **pt;
 Thread **st;
 cell_t c;
 Domain *pDomain = DOMAIN_SUB_DOMAIN(domain,P_PHASE);
 Domain *sDomain = DOMAIN_SUB_DOMAIN(domain,S_PHASE);
 real xc[ND_ND], y, x;
     mp_thread_loop_c (t,domain,pt)
          if (FLUID_THREAD_P(t))
            {
              Thread *tp = pt[P_PHASE];
              begin_c_loop (c,t)
                {
    C_CENTROID(xc,c,t);
    x=xc[0];
    y=xc[1];
     if ( y < 0.00292 + 0.0006*cos(6.283*(x-0.01945)/0.0778) )
     C_VOF(c,tp) = 1;
     else
     C_VOF(c,tp) = 0;
                }
         end_c_loop (c,t)
        }
              mp_thread_loop_c (t,domain,st)
          if (FLUID_THREAD_P(t))
            {
              Thread *sp = st[S_PHASE];
              begin_c_loop (c,t)
                {
```

```
        C_CENTROID(xc,c,t);
        x=xc[0];
        y=xc[1];
         if ( y < 0.00292 + 0.0006*cos(6.283*(x-0.01945)/0.0778) )
         C_VOF(c,sp) = 0;
         else
         C_VOF(c,sp) = 1;
                        }
             end_c_loop (c,t)
           }
}
```

국문초록

원자로에 있어서 열전달과 임계열유속(CHF)은 그 효율과 안전에 직결되므로 이를 향상시키는 연구는 주목 받아왔다. 가압경수로의 경우에도 비등의 문제는 사고시의 급격한 열전달 저하에 대응할 필요성에 있어서 주목할 수 밖에 없으며 핵융합로뿐만이 아니라 더 나아가 복합핵융합로 등을 위해서도 그러한 연구는 필요하다. 즉, 단순히 열전달의 능력의 문제보다는 사고시와 같은 다양한 경우를 고려한 열전달 설계가 필요하다. 따라서 설계에 있어서 단순함과 안정함, 지속가능성, 경제성과 관련이 되는 열전달 향상 및 임계열유속 향상을 위한 새로운 탐구가 필요하고 이를 이용한 설계방법론이 제시되어야 한다.

이러한 필요조건을 충족시키기 위하여 본 연구는 (국부적) 강제대류의 효과에서부터 시작하여 버블커트라는 개념을 제시한다. 이러한 개념을 바탕으로 계통도에 있어서 (국부적) 강제대류의 효과를 가져오는 형상설계의 필요성을 보여주고자 한다.

이를 위해, (국부적) 강제대류효과를 모델링 하기 위해서 ANSYS Fluent 의 VOF 모델을 사용하여 버블의 이탈속도와 Surface Nusselt Number 을 중심으로 그 열전달 증진양상을 살펴보았다.

이때 버블을 한쪽 방향으로 가열면에서 몰아내는 방안이 열전달효율을 높이며, 그 가열면에 강제로 밀어내는 유량이 클수록 표면 Nesselt Number 은 증가한다. 이때 효율적인 버블커트의 방법으로 막비등으로 발생한 버블을 가열면 한곳에 몰아넣고 그 가열면에 가까운 곳에 소량의 유체로 버블을 효과적으로 잘 이탈시키는 경우에도 열전달 증진의 효과는 많은 유량이 그 가열면에 쏘아질 때에 비해서 큰 차이를

보이고 있었다. 이를 이용한 원자로 설계기술은 응축기와 같은 곳에 많은 유량이 빠른 속도로 지나가는 것이 가장 효율적인 열전달 증진을 가져오는 것이기에 형상설계를 통해 베르누이 원리를 이용하여 열전달 증진이 필요한 부분의 유속을 증가시키는 방법이 제안된다.

이 논문에서는 이와 같이 혁신적인 열전달 향상 관련 기술의 개념을 제시하고 이를 적용하는 방법론에 대한 전산해석적인 확인을 바탕으로 열전달 향상을 도모하는 원자로 설계 개념에 대해 기술하여 이를 감안한 원자로설계에 도울 것이다.

핵심어: 열전달, 기포 거동, 기포 이탈, 원자로, 버블 커트, 국부적 강제순환, VOF

학 번: 2012-21010

감사의 글

원자력 발전소의 설계에 있어서 열전달 효율을 높게 만든다는 것은 어려운 일이고 저와 같은 이가 열전달 효율과 관련 있는 버블, 그런 주제로 논문을 쓰게 된 것은 영광입니다.

논문에서 6 장 내용으로 들어간 구조물로 버블을 잘라내는 방안이 2012 년 현대건설 기술상의 응모하여 그 실험결과를 알 수 없다는 논란 속에서 동상을 받게 된 것과 같은 행운에서 이 논문의 주제가 찾아졌습니다. 이러한 능력 밖의 주제에 대한 논문을 쓰는 시도는 여러 교수님들과 연구실 사람들의 격려와 도움이 있었기에 가능한 것입니다.

KNS 학술대회 참가자들 중 질문자는 저에게 전산해석 또는 실험을 하도록 주변에 도움을 청할 의사는 없는지 묻는 등 많은 분들이 무모한 주장을 하는 용기를 아껴주신 것 같습니다.

언제나 훌륭하고 어려운 일들과 좋은 논문 쓰는 것을 마치 강인한 기계처럼 척척 해내시는 서균렬 지도교수님과 열전달이라는 흥미로운 과목을 통해 만나 뵌 김응수 교수님, 다상유동의 어려운 분야를 연구하시며 이를 이용하여 저의 주장을 증명해보는 시도를 하는 길을 어렴풋이 보여주신 조형규 교수님께도 감사 드립니다. 논문 심사장에서 시간을 끄는 바람에 다음 심사를 위해 계시게 된 심형진 교수님과 주한규 학과장님께도 감사 드립니다.

급하게 논문의 내용을 실험기구를 사용하여 증명하고자 만나 뵈었고 많은 시간과 노력을 들여 실험기구 설계를 도와주신 태양전열의 전사장님께서 물심양면으로 도와주셨습니다. 감사 드립니다. 그러나 정작 실험을 하기도 어려운 남은 시간의 촉박함 속에서 손형민 박사님이 이 도전에 항상 함께하여 주어 천군만마를 얻은 격이 되어 감사 드립니다. 전산해석을 위해 컴퓨터를 사용하게 허락한 Palish 와 여러 도움을 준 Jubair 에게도 감사 드립니다.

곁에는 없지만 용기 있는 논문 쓰는 자세와 공학을 공부하는 모범적 자세를 보여준 Burhan 에게도 감사 드립니다. 저에게 청강을 허락해주신 기계항공공학부 교수님께도 감사 드립니다. 특히 항공기 또는 우주선의 형상에 따른 양력과 공기저항에 대해 전산해석방법을 가르치시는 교수님께서 기체역학에 대한 과목의 청강을 주셔서 짧은 2 년 동안 관련분야의 전산해석(조선분야 형상해석 포함)까지 한번 도전해보고 싶었던 계획에 마지막으로 불씨를 간직한 채 4 학기째를 보내게 된 것을 감사 드립니다. 그 존함을 타과의 학생인 저의 논문 감사 글에 쓰는 것이 옳은지를 몰라 그 존경하는 교수님을 들어내어 보이지는 못했습니다. 정말 어려운 과목이었습니다. 특히 그 교수님의 항공기의 공기저항에 대한 전산해석 수업이 저의 강의시간과 겹쳐 놓친 것이 아쉬움으로 계속 남을 것 같습니다.

건축토목관련 분야에 계획분야이지만 박사학위소지자라고 특혜(?)와 같은 배려로 수업을 진행하거나 청강을 허락하신 여러 교수님께 감사 드립니다. 제가 지방대학의 과거 전공관련 강의를 나가는 바람에 청강하는 과목에 결석에 많았음에도 핵융합분야의 나용수 교수님, 전산해석분야 주한규 학과장님, 그리고 여러 과목들에서 조형규 교수님, 김은희 교수님,

심형진 교수님께서 청강을 허락해 주신 것이 기억납니다. Oda 교수님의 전산재료해석분야는 저에게 흥미를 준 과목이었고, 현재 차관님이라고 불러야 할 이은철 교수님의 원전안전해석 과목도 배려 속에서 기초부터 배워갈 수 있어 감사 드립니다. 특별히 핵비확산 분야, 또는 핵주기 관련 기술 독자확보에 흥미를 느끼게 만들 기회에 감사 드립니다. 여러 강사님들께서 저에게 많은 자극이 되셨던 것으로 가슴에 간직하겠습니다.

끝으로 저의 아버지와 어머니께 감사 드립니다. 또 다른 분야에서 또다시 새로운 분야를 공부한다는 생각에 수긍하기란 쉽지 않은 일입니다. 2 년간 뒤돌아보면 그 도전력으로 여러 관심분야에서 기술개발능력을 확보하여 경제적인 가치창조도 쉽게 될 것이라고 여겼던 것이 삶에 있어서 어떠한 성과를 낼 수 있는 한계에 대한 저의 무지에서 출발했는지도 모르겠습니다. 이 논문이 전산해석 후 그 해석결과의 정리에 필요한 시간이 부족해 논문이 7 일의 99% 밤샘작업으로 급하게 쓰여지는 과정에서, 논문제출은 그 내용보완을 한 후에 제출하는 것이 가능하게 선택하지 않은 최선을 알아봐주신 여러 교수님들께 감사 드립니다. 이 논문이 6 장에서 일부 다룬 칼날을 이용해 버블을 끌어내겠다는 생각이 실험으로 증명되지 않았지만 저를 현대건설 기술상 장려상에 뽑아주신 당시 현대건설 위촉 심사위원님의 생각처럼 논문심사교수님들도 실제 증명되지 않았지만 혹시 있을 수 있는 가능성, 꿈에 점수를 후하게 주시는 실수(?)를 저지르신 것이 아닌지 하는 생각에 잠시 잠기게 됩니다. 하지만 현대건설 기술상에 내용의 15% 이상 한 20%를 인 일부를 차지했던 Swirl Pipe 로 열전달을 높이겠다는 방안은 2008 년부터 제가 주변에 발명할 수 있다고 떠들었기에 저의 발명품인줄로 착각을 했던 기억이 납니다. 그리고 2013 년 그 swirl 효과 관련 논문을 쓰려고 논문검색을 하니 2010 년 경에 누가 논문을 써버렸다는 것과 이것이 이미 핵융합로의 토카막에 사용되고 있다는 것과, 나용수 교수님 수업시간에도 소개되고 있다는 사실을 알게 되었습니다. 이와 같이, 어떤 가설이나 연구주제는 착상을 한 이가 하지 않으면 그의 동시에 똑같은 착상을 한 사람이나 그 주위의 또 다른 이들이 한다는 것을 뼈저리게 느끼게 되었습니다. 본 논문의 논의도 다른 이들이 수년 뒤에 곧 다루어볼 것만 같은 내용이기에 그 해석 깊이를 떠나 제가 분석을 해보았다는 것이 의미가 있을 것 같습니다. 더구나 버블커터의 효과나 Swirl pipe 효과는 동일한 형상설계 기법을 이용한 동일 artificial flow 인위적 형성이라는 동일 물리적 현상과 그 해석철학에서 출발하기에 그 절반을 직접 연구하며 다루었다는 결과가 되었습니다. Swirl pipe 를 개발하신 외국연구자들이 앞으로 그 철학의 또 다른 결과물을 만드는 것이 논문으로 읽게 된다고 해도 이제 담담히 박수 치며 격려할 수 있을 것 같습니다. 한 사고테마 연구한다는 것이 어려운 작업이라는 것 잘 느끼게 되었기에….

관악산에 찾아왔던 형과 형수님께도 감사 드립니다. 특히 어머니와 논문에 대해 많이 이야기를 한 것 같습니다. 그리고 하나님께 감사 드립니다.

모두에게 그리고 모든 상황에 감사 드립니다.